Advances in Engineering Software 101 (2016) 77–97

Contents lists available at ScienceDirect

Advances in Engineering Software

journal homepage: www.elsevier.com/locate/advengsoft

Comparison of different agglomeration multigrid schemes for compressible and incompressible flow simulations

Georgios N. Lygidakis, Sotirios S. Sarakinos, Ioannis K. Nikolos*

School of Production Engineering and Management, Technical University of Crete, 73100 Chania, Greece

ARTICLE INFO

Article history:
Available online 13 January 2016

Keywords:
Multigrid schemes
Isotropic/directional agglomeration
FAS
FMG
Parallel processing
Compressible/incompressible flow
Hybrid unstructured grids
RANS equations

ABSTRACT

Different parallel agglomeration multigrid schemes have been developed aiming to improve the computational performance of a compressible and an incompressible academic Computational Fluid Dynamics (CFD) codes, named *Galatea* and *Galatea-I*, respectively. Flow prediction is succeeded via the implementation of Reynolds-Averaged Navier–Stokes (RANS) equations combined with appropriate turbulence models on three-dimensional unstructured tetrahedral or hybrid meshes. The sequence of required coarser grids, composed of irregular polyhedral elements, is generated either with the isotropic or directional (full- or semi-coarsening) fusion of neighbouring control volumes on a topology-preserving framework; it resembles the advancing-front technique as it begins from solid wall surfaces and extends successively to the interior domain. The multigrid accelerated approximation of flow and turbulence equations is achieved via the V-cycle implementation of either the Full Approximation Scheme (FAS) or its coupled version with Full Multigrid (FMG) method. Multigrid approaches with different agglomeration and solution strategies have been extensively tested against three- and quasi-three-dimensional test cases, all of them demonstrating their potential for considerably improved efficiency. Their contributions to the reduction of simulations' computation time are analysed, while additionally the differences due to the type of the flow (compressible or incompressible) are thoroughly discussed.

1. Introduction

During the last decades three-dimensional unstructured grids have become an essential tool for Computational Fluid Dynamics (CFD), extending its applications to complex geometries. However, despite the corresponding solvers are enhanced with the largest possible flexibility in the treatment of such complicated geometries, along with the minimum user interaction for their generation/adaptation, they appear to be relatively inferior in terms of efficiency compared to the structured ones [1–6]. A remedy to this considerable deficiency is revealed to be the multigrid methodology, a scheme originally proposed to increase the convergence rate of the numerical solution of elliptic problems [7,8], but since then it has made its way to various types of numerical simulations [4,5,9–15]. Its main idea derives from the observation that most of the well-established iterative methods converge more slowly on finer grids [8], as they succeed in time-effective relaxation of high frequency errors but they seem to be relatively inefficient against the low frequency ones [4,6]; the more denser resolution is used the more slower damping of low frequency errors is achieved. The multigrid technique, which is based on the solution of the governing Partial Differential Equations (PDE's) on successively coarser grids, actually transforms these low frequency errors in high frequency ones on the coarser resolutions, allowing for their efficient damping [4]. The solution produced at each coarser mesh is combined with this of its finer one for accuracy reasons; therefore, appropriate associating relations, namely restriction and prolongation operators, are required to be defined between each two successive grids [6]. Based on the aforementioned main concept of multigrid methodology, various versions of it have been developed during the past years; their differences are mainly focussed on the way the sequence of spatial resolutions is generated as well as on the associating relations used between each two successive grids [1–4,8,16–23].

Regarding the generation of coarser grids, the multigrid method can be divided in two main types, namely the geometrical and the agglomeration one [1,4,6]. According to the geometrical-type schemes, the sequence of the coarser resolutions can be derived either by the generation of completely independent grids from the very beginning (non-nested approach) or by the construction of associated (nested) grids; the latter is performed by beginning either from the coarsest one and enriching it with a refinement method [24,25] or from the densest one by removing Degrees of Freedom (DoF's) and implementing re-triangulation [1,4]. Similarly to the

* Corresponding author. Tel.: +30 28210 37300; fax: +30 28210 06900.
E-mail address: jnikolo@dpem.tuc.gr (I.K. Nikolos).

http://dx.doi.org/10.1016/j.advengsoft.2015.12.004
0965-9978/

last technique, the agglomeration multigrid approach, introduced by Lallemand [26], considers the construction of coarser grids from the initial finest one via the isotropic fusion of neighbouring control volumes; as a result, coarser meshes with irregular polyhedral elements are produced [1,4,6]. Despite the satisfactory acceleration entailed by this scheme for inviscid flow problems along with tetrahedral grids, its reduced performance is observed for viscous flow simulations performed on hybrid grids [16,20,27]. A semi-coarsening or directional agglomeration technique was proposed by Mavriplis [16] to alleviate the effects of this deficiency. According to this approach the control volumes of nodes belonging to stretched elements are merged only if they are aligned with the normal to the boundary surface direction; for the rest ones the standard isotropic procedure is followed. In that way the mesh anisotropy is moderated as the generated coarser grids appear to be less stretched than the initial finest one [27]. Alternatively, full-coarsening directional agglomeration [19-21,28] can be employed in such regions, according to which the procedure begins by merging the boundary control cells, while at next a line-agglomeration step is performed, by fusing control volumes along implicit lines starting directly above the boundary volumes [20]; a deeper agglomeration compared to semi-coarsening is achieved, preserving though the topology of the initial grid. Besides the aforementioned topological multigrid types, another popular type is the Algebraic Multigrid (AMG), which considers the construction of a coarsening matrix rather than the generation of any new grid [1,8].

The second characteristic distinguishing multigrid methods, concerns the way the data are exchanged between each two successive grids. The initially developed algorithms were employing the Full Multigrid scheme (FMG) [1,8], according to which, since the fully or partially relaxed solution on the coarsest grid is obtained, it is interpolated (prolongated) to the finer one and used as an initial guess. The same procedure is repeated up to the finest resolution, succeeding in that way a cheaper initial condition than the usually utilized unphysical uniform one [8,29]. Alternatively, the Full Approximation Scheme (FAS) can be implemented; at each multigrid cycle it considers the solution of the governing PDE's only for the finest resolution, while it employs approximate versions of them for the coarser ones [6]. Since relaxation is completed on the finest mesh, the values of variables and flux balances are transferred (restricted) to the coarser one [4]. The aforementioned procedure is repeated up to the coarsest grid, while at next the computed variables' corrections are interpolated (prolongated) successively back to the finest resolution, accomplishing in that way a V-cycle process [6]; otherwise a W-cycle strategy can be followed [1]. A combined (nested) FMG–FAS approach was studied by Lambropoulos et al. [27], according to which the FAS process is incorporated in the FMG one; the PDE's are relaxed, beginning from the coarsest grid (preliminary stage) and, as the number of iterative cycles increases, the FAS extends successively to the finer meshes up to the finest one, at which point the main stage begins [6,27].

In this work the development of a parallel agglomeration multigrid methodology to accelerate compressible and incompressible fluid flow simulations is initially reported. It is based upon a previous study of the authors [6], including though further advances of the incorporated multigrid scheme, besides a more detailed description of it, e.g., extra and slightly different constraints for limiting the fusion of control cells at solid wall surfaces and lower prismatic layers are imposed, while a distance-based prolongation process is used in case of viscous flows. In general, agglomeration of adjacent control volumes is performed in a way analogous to the advancing front technique, as the whole procedure begins with the fusion of the viscous boundary nodes' control cells at each sub-domain (in which the initial grid is divided for parallel processing) and then extends to the internal ones. However, the fusion can be performed in isotropic, semi- or full-coarsening directional mode, depending on the flow type and consequently on the grid type [4,6,29]. Special care is required for the *ghost* nodes at the overlapping regions between adjacent sub-grids [4], to be merged or remain singletons, according to the agglomeration of their corresponding *core* nodes at neighbouring partitions [6,29]. In order to achieve a multigrid accelerated iterative solution, either the FAS [6] or the combined FMG–FAS [27] procedure is followed, the latter dividing the whole procedure in a preliminary and a main stage [27]. For the evaluation of the proposed methodology compressible code *Galatea* [29,30] and incompressible code *Galatea-I* [6] are used. These relatively recently developed, node-centred finite-volume solvers employ the appropriate Reynolds-Averaged Navier–Stokes (RANS) PDE's [30-34] along with suitable two-equation turbulence models, namely k–ε [35,36], k–ω [37] and SST (Shear Stress Transport) [38] (for *Galatea-I* only SST), on hybrid unstructured grids including tetrahedral, prismatic and pyramidical elements. Artificial compressibility (or pseudo-compressibility) methodology is utilized in *Galatea-I*, which adds a temporal derivative of pressure to the continuity equation allowing in that way incompressible PDE's to be solved within the framework of a time-marching compressible flow algorithm [1,8,39,40]. Both algorithms are further accelerated with parallel processing, based on domain decomposition approach and Message Passing Interface (MPI) library functions [41-45]. The proposed multigrid methodology was validated against available in the literature compressible and incompressible benchmark test cases; the obtained results demonstrate the improvement of codes' computational performance with its implementation. Moreover, they reveal the differences among the applied multigrid alternatives as well as those due to the type of the flow (compressible or incompressible).

2. Flow model

2.1. Governing equations

The compressible (Favre-averaged) or incompressible (Reynolds-averaged) Navier–Stokes equations (RANS) are described in three-dimensional differential formulation as [1]

$$\frac{\partial \vec{W}}{\partial t}+\frac{\partial \vec{F}^{inv}}{\partial x}+\frac{\partial \vec{G}^{inv}}{\partial y}+\frac{\partial \vec{J}^{inv}}{\partial z}-\frac{\partial \vec{F}^{vis}}{\partial x}-\frac{\partial \vec{G}^{vis}}{\partial y}-\frac{\partial \vec{J}^{vis}}{\partial z}=\vec{S} \tag{1}$$

where $\vec{S}$ denotes the vector of the source term, equal to zero in this study. The flow variables' vector $\vec{W}$ is defined for compressible (five conservative variables) and incompressible (four primitive variables) flow as [6, 30]

$$\vec{W}_{compressible}=\begin{bmatrix}\rho & \rho u & \rho v & \rho w & \rho E\end{bmatrix}^T \tag{2}$$

$$\vec{W}_{incompressible}=\begin{bmatrix}p & u & v & w\end{bmatrix}^T \tag{3}$$

where ρ is the density, u, v, w the velocity components, p the pressure and ρE the specific total energy, all in dimensionless formulation; for incompressible flow the energy equation is not included. The vectors $\vec{F}^{inv}$, $\vec{G}^{inv}$, $\vec{J}^{inv}$ and $\vec{F}^{vis}$, $\vec{G}^{vis}$, $\vec{J}^{vis}$ are the inviscid and viscous flux terms respectively, described in case of compressible flow as [1, 30]

$$\underbrace{\begin{pmatrix}\rho u\\ \rho u^2+p\\ \rho uv\\ \rho uw\\ (\rho E+p)u\end{pmatrix}}_{\vec{F}^{inv}},\ \underbrace{\begin{pmatrix}\rho v\\ \rho vu\\ \rho v^2+p\\ \rho vw\\ (\rho E+p)v\end{pmatrix}}_{\vec{G}^{inv}},\ \underbrace{\begin{pmatrix}\rho w\\ \rho wu\\ \rho wv\\ \rho w^2+p\\ (\rho E+p)w\end{pmatrix}}_{\vec{J}^{inv}} \tag{4}$$